

Institute for Computational Mechanics in Propulsion (ICOMP)

Eleventh Annual Report - 1996

Editors:

Theo G. Keith, Jr.
Karen Balog
Louis A. Povinelli

May 1997

CONTENTS

	Page
INTRODUCTION	1
THE ICOMP STAFF OF VISITING RESEARCHERS	2
RESEARCH IN PROGRESS	9
REPORTS AND ABSTRACTS	33
SEMINARS	39
COOLANT FLOW MANAGEMENT WORKSHOP	45

**INSTITUTE FOR COMPUTATIONAL MECHANICS IN PROPULSION
(ICOMP)**

ELEVENTH ANNUAL REPORT

1996

SUMMARY

The Institute for Computational Mechanics in Propulsion (ICOMP) was formed to develop techniques to improve problem-solving capabilities in all aspects of computational mechanics related to propulsion. ICOMP is operated by the Ohio Aerospace Institute (OAI) and funded via a cooperative agreement by the NASA Lewis Research Center in Cleveland, Ohio. This report describes the activities at ICOMP during 1996.

INTRODUCTION

The Institute for Computational Mechanics in Propulsion (ICOMP) was established at the NASA Lewis Research Center in September 1985. The overall purpose was to improve problem-solving capabilities in all aspects of computational mechanics relating to propulsion. ICOMP provides a means for researchers with experience and expertise to spend time in residence at Lewis performing research to improve computational capability in the many broad and interacting disciplines of interest in aerospace propulsion.

The scope of the ICOMP program is to advance the understanding of aerospace propulsion physical phenomena and to improve computer simulation of aerospace propulsion systems and components. The specific areas of interest in computational research include: fluid mechanics for internal flows; CFD methods; turbulence modelling; and computational aeroacoustics.

This report summarizes the activities at ICOMP during 1996.

The following sections of this report provide lists of the resident and visiting researchers, their affiliations and educational backgrounds. Individual sections are provided which briefly describe reports of RESEARCH IN PROGRESS, the REPORTS AND ABSTRACTS published over the past year and the SEMINARS presented throughout the year. The agenda and overview of a very productive workshop held in 1996 are also given. The 1-1/2 day event entitled "Cooling Flow Management Workshop" was held December 12-13, 1996.

THE ICOMP STAFF OF VISITING RESEARCHERS

The ICOMP research staff for 1996 is shown in Table I. A total of thirty-three researchers were in residence at ICOMP for periods varying from a few days to a year. The resident staff numbered twenty while the visiting staff, including one graduate student, numbered thirteen.

As usual, the resident researchers were very productive Table II provides a numerical summary of ICOMP during its eleven years of operation in terms of research staff size and technical output as measured by the numbers of seminars, reports and workshops.

Table I. - The ICOMP Research Staff-1996

A. Resident Staff.

Kumud Ajmani, Ph.D., Mechanical Engineering, Virginia Polytechnic Institute and State University, 1991. January, 1992–Present.

Hannes Benetschik, Ph.D., Mechanical Engineering, TU Aachen Institute, 1992. November, 1995–November, 1996.

Joongkee Chung, Ph.D., Mechanical Engineering, University of California, Berkeley, 1991. May, 1992–Present.

Datta Gaitonde, Ph.D., Mechanical and Aerospace Engineering, Rutgers University, 1989. September, 1995–Present.

Duane R. Hixon, Ph.D., Aerospace Engineering, Georgia Institute of Technology, 1993. October, 1993–Present.

Bo-nan Jiang, Ph.D., Engineering Mechanics, University of Texas, Austin, 1986. October, 1987–Present.

Kai-Hsiung Kao, Ph.D., Aerospace Engineering Sciences, University of Colorado, 1989. November, 1992–Present.

William W. Liou, Ph.D., Aerospace Engineering, Pennsylvania State University, 1990. November, 1990–December, 1996.

James M. Loellbach, Ph.D. expected 1995, Aeronautical and Astronautical Engineering, University of Illinois. May, 1992–Present.

Andrew T. Norris, Ph.D., Mechanical and Aerospace Engineering, Cornell University, 1993. June, 1993–Present.

Aamir Shabbir, Ph.D., Mechanical Engineering, State University of New York, Buffalo, 1987. June, 1991–Present.

Shyue-Horng Shih, Ph.D., Aerospace Engineering, University of Cincinnati, 1993. June, 1993–Present.

Tsan-Hsing Shih, Ph.D., Aerospace Engineering, Cornell University, 1984. March, 1990–Present.

Erlendur Steinthorsson, Ph.D., Mechanical Engineering, Carnegie Mellon University, 1992. January, 1992–Present.

John Slater, Ph.D., Aerospace Engineering, Iowa State University, 1992. February, 1996–Present.

Gerald Trummer, B.S., Michigan State University, 1982. September, 1995–Present.

Fu-Lin Tsung, Ph.D. expected 1995, Aerospace Engineering, Georgia Institute of Technology. March, 1993–Present.

Wu, Jie, Ph.D., Civil Engineering, University College of Swansea, 1993. March, 1993–October 1996.

Shaye Yungster, Ph.D., Aeronautics and Astronautics, University of Washington, 1989. November, 1989–Present.

Jiang Zhu, Ph.D., Mechanics, Institut National Polytechnique de Grenoble, 1986. April, 1992–May 1996.

B. Visiting Staff/Consultants.

Jack Edwards, Ph.D., Aerospace Engineering, North Carolina Agricultural and Technical State University, 1993. Research Associate, NASA Center of Research Excellence, North Carolina A&T State University.

Thomas Hagstrom, Ph.D., Applied Mathematics, California Institute of Technology, 1983. Associate Professor, Department of Mathematics and Statistics, University of New Mexico.

Arne Johansson, Ph.D., Department of Mechanics, The Royal Institute of Sweden, Stockholm, Sweden, 1983. Professor and Chairman, Department of Mechanics, The Royal Institute of Sweden, Stockholm, Sweden.

Duo-Min Lin, Ph.D., Division of Engineering and Applied Science, California Institute of Technology, 1996.

R. R. Mankbadi, Ph.D., Engineering, Brown University, 1979. Professor, Engineering, Cairo University.

Alexander Oron, Ph.D., Mechanical Engineering, Technion-Israel Institute of Technology, 1986. Senior Lecturer, Department of Mechanical Engineering, Technion-Israel Institute of Technology.

Richard H. Pletcher, Ph.D., Mechanical Engineering, Cornell University, 1966. Professor, Department of Mechanical Engineering, Iowa State University.

Oleg Ryzhov, Ph.D. Rensselaer Polytechnic Institute Computing Center, USSR Academy of Sciences, Moscow.

Gretar Tryggvason, Ph.D., Engineering, Brown University, 1985. Associate Professor, Department of Mechanical Engineering and Applied Mechanics, University of Michigan.

Eli Turkel, Ph.D., Applied Mathematics, New York University, 1970. Professor, Department of Mathematics, Tel Aviv University, Tel Aviv, Israel.

Xuesong Wu, Ph.D., Department of Mathematics, Imperial College.

Akira Yoshizawa, Sc. D., Physics, University of Tokyo, Tokyo, Japan, 1970. Professor, Institute of Industrial Science, University of Tokyo.

Table II. - ICOMP STATISTICS (1986 TO 1996)

	1986	1987	1988	1989	1990	1991	1992	1993	1994	1995	1996
RESEARCHERS	23	43	50	46	47	49	58	64	50	30	33
SEMINARS	10	27	39	30	37	26	32	46	3	15	10
REPORTS	2	9	22	32	25	29	27	51	32	28	13
WORKSHOP/ LECT. SERIES	1	0	2	1	1	1	1	1	2	2	1
NO. OF PRESENTATIONS	7	0	21	14	15	21	15	33	40	23	

Research in Progress

RESEARCH IN PROGRESS

KUMUD AJMANI

Research Area: Development of Codes for Parallel Processing

State-of-the-art inlet design is one of the key requirements of the NASA High Speed Research (HSR) program and the High Speed Civil Transport (HSCT) programs. The role of the inlet is to decelerate the air entering the engine from the air-speed to about Mach 0.4, since this is the optimal air-speed for efficient compressor performance. Reduction of circumferential distortion (of total-pressure) and optimization of total-pressure recovery are two of the primary issues in inlet design.

In order to facilitate inlet design for the HSCT program, extensive experimental tests have been performed with a fully subsonic transition duct geometry at the NASA Lewis Research Center. One of the goals of Dr. Ajmani's past year's work has been to perform a detailed comparison of this experimental data with computational results. The objective of performing such a study is to develop an inlet-design knowledge base, which can be used to test candidate HSCT inlet geometries. A secondary objective is to study the effect of vortex generators in reducing circumferential distortion at the outflow face of the inlet, i.e., at the compressor inflow face.

The CFD computations have been performed with a parallel, CHIMERA overset-grid code (in collaboration with Dr. Meng-Sing Liou of NASA LeRC and Dr. Kai-Hsing Kao of ICOMP). The agreement between experimental data and computations has been satisfactory - however, the performance of the Baldwin-Lomax model in predicting turbulence quantities has been less than satisfactory. In order to alleviate this deficiency, work has been initiated to program a better, $k-\epsilon$ turbulence model into the overset grid code (in collaboration with Dr. T.-H. Shih of ICOMP). The effect of vortex generators on inlet performance will be revisited through the CFD simulations, after the new turbulence model is tested and validated.

Some of his work during this year has also focussed on furthering the development of parallel-computing strategies for the CHIMERA overset-grid code. The results indicate that the performance of the parallel, message-passing version is very competitive with the performance obtained from a Cray C-90 implementation. This was verified by running various tests with a 16-processor cluster of IBM RS-6000 workstations (LACE cluster at NASA Lewis), and a 64-processor Cray T-3D machine.

HANNES BENETSCHIK

Research Area: Turbomachinery and Grid Generation Codes

During this year, part of Dr. Benetschik's research involved work on the quasi- and fully three-dimensional Euler- and Navier-Stokes codes EUNSS1/3D. The initial Q3D code, of which the development of dates back to his individual research, has been extended to fully 3D capabilities with former co-workers at the Technical University Aachen.

Dr. Benetschik's work at ICOMP has been focussed on numerical investigations involving Roe's Riemann solver combined with van Leer's MUSCL-technique. The latter was set up with various sets of dependent quantities, i.e., conserved, primitive and characteristic quantities. Primitive quantities have been found to provide the most smooth set of interpolants while maintaining the accuracy of the

conserved quantities, which can be theoretically proven to be the most accurate choice. Subsequently, the implementation of a Low-Reynolds number $k-\epsilon$ turbulence model has been improved. The work was focussed on deriving numerically well-conditioned source-Jacobians and on advising clipping criteria preventing the turbulence quantities from assuming nonphysical values.

The objective of the second field of CFD-research involved transition modeling. The studies are based on the comprehensive analyses of Mayle and Gostelow. Various correlations suggested in their work have been implemented in the Q3D-code EUNSS1, which employs the algebraic turbulence model of Baldwin and Lomax. The latter has been refined by the suggestions of Chima, Giel and Boyle, 1993. Numerous computations have proven that transition modeling has to take into account several modes. The usual assumption of bypass transition at turbulence intensities, which are typical for turbomachinery is insufficient. Even at turbulence intensities beyond 5%, transition may be induced by laminar separation. Thus, separation induced transition may prevent the flow from separating globally under adverse pressure gradient conditions. In the present investigation, the set of correlations which has been implemented in the code which allows for capturing both modes mentioned, were in good agreement with experimental data that has been found.

Publications

H. Benetschik, A. Lohmann, J.R. Luecke, and H.E. Gallus: "Inviscid and Viscous Analysis of Three-Dimensional Turbomachinery Flows Using an Implicit Upwind Algorithm". AIAA Paper 96-2556, 11 pages.

J.R. Luecke, W. Sanz, H. Benetschik, H.E. Gallus, and M.F. Platzer: "Numerical Investigation of Transition and Hub Corner Stall Phenomena inside an Annular Compressor Cascade". AIAA Paper 96-2655, 8 pages.

H. Benetschik, and C. Hah: "Towards the Prediction of Transition in Turbomachinery Cascades Using a Modeling Approach". Submitted for the 28th AIAA Fluid Dynamics Conference, Snowmass, Co. 1997, 10 pages.

JOONGKEE CHUNG

Research Area: Code Development for Unsteady Inlet Flows Using Parallel Processing

An evaluation of the Compressor Face (CF) boundary conditions for a supersonic inlet was performed using a Variable Diameter Centerbody geometry. Several comparisons of the predictions were made with existing experimental data. The CF boundary conditions, using a radially varying static pressure, were applied in the viscous computations of a dynamic response study and revealed an oscillatory behavior. These oscillations were not predicted in inviscid computations. The turbulence model, or the existence of vortex generators may have played roles in the differences. Further study is needed to verify the cause of the disturbance. Coarse grids were used to reduce the enormous CPU requirements.

Development of a bleed/bypass boundary condition was performed after the assessment of currently used boundary conditions and computations with an inlet which had physically attached centerbody bleed slots. The initial stage of coding work was done for a choked bleed boundary by incorporating available experimental data in a "table look-up" format which has more geometry variations for the slot-type bleed as well as porous bleed.

A coordinated effort to incorporate new algorithms and boundary conditions developed in the past into the official version of NPARC (v3.0) has initiated. This modified version of NPARC will continue to support controls CFD interdisciplinary research and NPARC/ADPAC code coupling study using Visual Computing Environment (VCE) being developed by CFD Research Corporation.

JACK EDWARDS

Research Area: Low Velocity Preconditioning

During Dr. Edwards visit to ICOMP, he worked with Dr. Meng-Sing Liou (IFMD) on extending low-diffusion upwind schemes (AUSM, AUSM+, AUSMDV, AUSM-W) to operate effectively at very low speeds. The extensions are designed for use with time-derivative preconditioning and are constructed using the eigenvalues of the preconditioned Euler system. A key element of the extension is the recognition that, at very low velocities, the physical speed of sound should not be used to scale the artificial diffusion inherent in the upwinding formulations. By employing the preconditioned eigenvalues, this unphysical scaling is avoided, and the result is a robust, accurate flux-splitting methodology suitable for flows at all speeds. The low-diffusion properties of the baseline schemes are preserved, and the new extensions blend smoothly into the original formulations as the local Mach number approaches unity. Steps toward developing a robust implicit integration strategy for the new schemes were also undertaken, as was an extension to multi-component gas mixtures. The results of this research have been submitted for presentation at the 13th AIAA Computational Fluid Dynamics conference.

DATTA GAITONDE

Research Area: Electromagnetics

High-order compact difference-based schemes were combined with the finite-volume formulation to investigate wave propagation phenomena with particular emphasis on computational electro-magnetics (CEM) in the time domain. The formulation combined the primitive function approach of ENO schemes^[1] with a five-point compact difference stencil. In addition to a sixth order scheme, Fourier optimization techniques (see [2] or [3] for example) were employed to develop a single-parameter family of fourth-order methods by minimizing semi-discrete dispersion and isotropy errors^[4]. The scheme was formally extended to multiple-dimensions^[5]. The error incurred with the transformed domain approach on stretched meshes was quantified by evaluating the truncation error. Physical space differencing formulas were developed for use on stretched meshes^[5].

The full discretization employed the classical fourth-order Runge-Kutta method for time integration. Stability bounds were established for the spectrum of schemes and the optimum coefficients for the spatial discretization were rederived for specific time-step sizes. Consistent boundary conditions were developed for both the compact derivative of the primitive function as well as for the physical fluxes^[4,5]. To enforce stability in certain situations, up to tenth order symmetric compact filter formulas were derived and analyzed^[5]. The scheme was employed successfully on several configurations including i) propagation of a transverse electromagnetic TEM polarized pulse in an infinite medium, ii) a boundary-dominant problem consisting of 1-D cavity formed by a pair of closely spaced perfect electrical conductors (PEC) iii) a 3-D rectangular wave guide, iv) a 3-D spherical dipole field and v) scattering of a Gaussian pulse by a sphere for radar cross-section analysis. In each case, the compact five-point stencil operators were seen to be far superior to five-point explicit, three-point compact and MUSCL schemes.

Previous endeavors focused on shock-wave/turbulent boundary layer interactions were successfully transitioned to archival publication^[6-8]. Present efforts include examining the evolution of flow field structure in a crossing-shock geometry under increasing interaction strength and asymmetry. In addition, the above noted algorithms developed for CEM application are being adapted for CFD.

1. Harten, A., Engquist, B., Osher, S. and Chakravarthy, S.R., "Uniformly High Order Accurate Essentially Non-oscillatory Schemes, III", *J. Comput. Phys*, Vol. 71, 1987, p 231.
2. Lele, S.K., "Compact Finite Difference Schemes with Spectral-like Resolution", *J. Comput. Phys*, Vol 103, 1992, p 16.
3. Chen, C.L. and Liu, Y., "Analysis of Numerical Approaches for Acoustic Equations", AIAA Paper 93-4324, 1993.
4. Gaitonde, D. and Shang, J.S., "High-Order Finite-Volume Schemes in Wave Propagation Phenomena", AIAA Paper 96-2335, 1996.
5. Gaitonde, D., Shang, J.S. and Young, J.L., "Practical Aspects of High-order Accurate Finite-Volume Schemes for Electromagnetics", AIAA Paper 97-0363, 1997.
6. Gaitonde, D. Edwards, J.R. and Shang, J.S., "Performance of Eddy-Viscosity Based Turbulence Models in a 3-D Turbulent Interaction", *AIAA Journal*, Vol. 34, No. 4, 1996, p 1590.
7. Gaitonde, D. and Shang, J.S., "Skin-Friction Predictions in a Crossing-Shock Turbulent Interaction", accepted for publication, *AIAA Journal of Propulsion and Power*.
8. Gaitonde, D. Edwards, J.R. and Shang, J.S., "The Structure of a Supersonic 3-D Cylinder/Offset-Flare Turbulent Interaction", accepted for publication, *AIAA Journal of Spacecraft and Rockets*.

THOMAS HAGSTROM

Research Area: Algorithms for Boundary Layer Value Problems, Domain Decomposition

Research work during FY96 involved two separate efforts:

High-Order Radiation Boundary Conditions

The issue of boundary conditions at artificial boundaries is crucial to the accurate simulation of long time wave dynamics, and hence of great importance to the development of computational aeroacoustics and electromagnetics. The work is focussed on the development of sequences of conditions which in some sense converge, with increasing order, to the "exact" or "transparent" condition. Issues to be addressed include both the practical implementation of such conditions, their mathematical analysis (in particular expected convergence rates), and their extension to more complicated systems.

We have considered two distinct but related sequences of approximate conditions, applied to the wave equation, Maxwell's equations, and the linearized Euler equations of gas dynamics. In joint work with J. Goodrich of NASA-Lewis, we are considering the classical Padé approximants to the symbol of the exact condition while with S.I. Hariharan of ICOMP we are considering conditions derived from progressive wave expansions. Recent progress includes:

- a) A complete characterization of the stability of sequences of conditions based on Padé approximants of the symbol in the case of the linearized Euler equations.
- b) A successful implementation of the Padé inflow conditions based on the new stability analysis. (Successful implementations of outflow conditions were previously achieved.)
- c) A start towards understanding corner compatibility conditions in the Padé case.
- d) A new formulation of sequences of conditions for the wave equation based on progressive wave expansions along with computational experiments.
- e) A framework for the development of progressive wave expansions and asymptotic boundary conditions for general constant coefficient hyperbolic systems.
- f) Application of the above to the linearized Euler equations, along with computational tests involving quadrupole sound sources near a vortex dipole.

Numerical Computation of Flame Speed and Structure

The accurate simulation of combustion phenomena is of great importance in the development of propulsion systems. Dr. Hagstrom jointly with K. Radhakrishnan of NYMA, have a long term project to create a combustion simulator combining high-order numerical methods and the capability to include complex reaction and diffusion physics. A first, but important, stage in this project is the production of a code to compute flame speed and structure, with an emphasis on robustness, i.e., relative insensitivity to initial approximations to temperature and species profiles. The technique combines time-stepping with an automatic determination of a stabilizing inflow rate and convergence acceleration based on inexact Newton or secant iterations. In the past year they have added a number of new features to the code and carried out more tests. Particular areas of progress include:

- a) An analysis of the stability of the "control" problem used to compute the flame speed. This lead to new flame speed corrections which somewhat improve convergence properties.
- b) Testing of convergence accelerators based on Broyden's method and on Newton-GMRES both combined with line searches. In particular they found that the latter method was quite successful in improving the code's convergence behavior.
- c) Testing with new fuels, including propane.
- d) Inclusion of more complex diffusion velocity models, using conjugate gradient methods to interactively solve the equations for these velocities.

DUANE R. HIXON

Research Area: Aeroacoustics

Work continued in the prediction of supersonic jet noise using the linearized Euler equations. Efforts focused on improving the performance of the numerical algorithm used, as well as performing parametric studies on more complex mean flows.

The existing solver used the 2-4 scheme of Gottlieb and Turkell^[1]. This solver requires approximately 25 points per wavelength and a time step of $CFL = 0.25$ to propagate acoustic waves correctly. Investigation into the properties of the 2-4 scheme yielded a new class of MacCormack-type schemes with much improved performance^[2]. Application of Tam and Webb's Dispersion Relation Preserving methodology^[3] and Hu, et. al.'s Low Dissipation and Dispersion Runge-Kutta method^[4] to the new scheme allows as few as 8 points per wavelength with a time step of $CFL = 1.25$.

This new scheme was evaluated using the benchmark problems of the first CAA Workshop^[5], and was used to solve the more difficult problems in the second CAA Workshop^[6]. At this point, the code was applied to the supersonic jet noise test cases, and was validated against the previously validated 2-4 scheme's results. The code was then applied to the parametric study of noise generated in a coannular jet exhaust^[7].

During the next year, the code will be revised to run on workstations, and a parallel implementation of the code will be performed. The code will be applied to the prediction of broadband noise ($0.025 < St < 1.5$) from a supersonic jet using a mean flow given by NPARC and comparing with experimental and analytical data. The new scheme will also be evaluated for use in the nonlinear Large Scale Simulation code.

References

1. Gottlieb, D. and Turkel, E. "Dissipative Two-Four Method for Time Dependent Problems", *Mathematics of Computation*, Vol. 30, No. 136, 1976, pp. 703-723.
2. Hixon, R. "On Increasing the Accuracy of MacCormack Schemes for Aeroacoustic Applications", AIAA Paper 97-1586, May 1997. Submitted to *J of Comp. Phys.*
3. Tam, C. K. W. and Webb, J. C., "Dispersion-Relation-Preserving Finite Difference Schemes for Computational Acoustics", *J. Comp. Phys.*, Vol. 107, 1993, pp. 262-281.
4. Hu, F. Q., Hussaini, M. Y., and Manthey, J., "Low-Dissipation and -Dispersion Runge-Kutta Schemes for Computational Acoustics", *J. Comp. Phys.*, Vol. 124, 1996, pp. 177-191.
5. Hixon, R. "Evaluation of a High-Accuracy MacCormack-Type Scheme Using Benchmark Problems", submitted to *Journal of Computational Acoustics*.
6. Hixon, R., Shih, S.-H., and Mankbadi, R. R., "Application of an Optimized MacCormack-Type Scheme to Acoustic Scattering Problems", to appear in the Proceedings of the Second CAA Workshop, Tallahassee, FL, Nov. 1996.
7. Hixon, R., Shih, S.-H., and Mankbadi, R. R., "Effect of Coannular Flow on Linearized Euler Equation Predictions of Jet Noise", AIAA Paper 97-0284, Jan. 1997.

BO-NAN JIANG*Research Area: Flow Applications of the Least-Squares Finite Element Method*

Dr. Jiang continued work with Jie Wu on the development of the least-squares finite element method (LSFEM). This method is based on minimizing the residuals of a first-order system of differential equations. The LSFEM is a simple and universal method for the numerical solution of all types of partial differential equations. The LSFEM and the continuum surface force method was combined to solve several incompressible multi-phase flow problems. Various benchmark tests have been carried out for both two-dimensional planar and axisymmetric flows, including a dam breaking, oscillating and stationary bubbles and a conical liquid sheet in a pressure swirl atomizer. The LSFEM has been applied to electromagnetic scattering and Radar Cross Section (RCS) calculations. In contrast to most existing numerical procedures, in which the divergence-free constraints are omitted, the LSFEM directly incorporates two divergence equations in the discretization process. The importance of including the divergence equations is demonstrated by showing that otherwise spurious solutions with large divergence will occur near the scatterers. Also a theoretical analysis was performed to show that the origin of spurious solutions in computational electromagnetics, which violates the divergence equations, is not due to the numerical process, but is deeply rooted in a misconception about the first-order Maxwell's equations and in an incorrect derivation and use of the curl-curl equations.

References

Jiang, B.N., Wu, J. and Povinelli, L.A. (1996): "The Origin of Spurious Solutions in Computational Electromagnetics", *J. Comput. Phys.* **125**, 104-123; see also NASA TM 106921, ICOMP-95-8.

Wu, J. and Jiang, B.N. (1996): "A Least-Squares Finite Element Method for Electromagnetic Scattering Problems", NASA Contractor Report 202313, ICOMP-96-12.

Wu, J., Yu, S.T. and Jiang, B.N. (1996): "Simulation of Two-Fluid Flows by the Least-Squares Finite-Element Method using a Continuum Surface Tension Model", NASA Contractor Report 202314, ICOMP-96-13.

ARNE JOHANSSON*Research Area: Algebraic Reynolds Stress Models*

The research contacts between the Department of Mechanics at the Royal Institute of Technology in Sweden and ICOMP have been established for several years. The present visit involved discussions and analyses of particularly Explicit Algebraic Reynolds Stress Models (EARSIM) and their application to three-dimensional flows. Flows with swirl, such as the axially rotating pipe flow, were given special attention. New software development tools using automated code generation with software algebra tools were demonstrated by T. Sjogren. These new tools include user-friendly interfaces and have been developed within the Department of Mechanics. A seminar on these topics was given by Professor Johansson in which he also demonstrated the software tools. Areas of interest for future exchange between Professor Johansson's and Dr. Shih's groups were discussed and include, but are not limited to, Modelling of Scalar Transport.

KAI-HSIUNG KAO

Research Area: 3D Compressible Finite Volume Navier-Stokes Flow Solver

The majority of Dr. Kao's research this year was in the area of coupled aero/thermal/structure analysis (CATS) of the secondary flow system in an air-breathing engine. He was generally engaged in developing and demonstrating the CFD capability for predicting the flowfields inside the rotating drum within the H.P. compressor of the Pratt & Whitney 4000 engine. An accurate numerical algorithm (AUSM) combined with an efficient and flexible grid generation technique (Chimera or DRAGON) was used. A turbulent model which was developed within the ICOMP turbulence group has been implemented and validation of the model is currently being monitored.

In addition, a hybrid Chimera/unstructured grid system has been developed to enhance grid flexibility so that the conjugate fluid and solid body regions can be studied simultaneously. As a result, the multidisciplinary analysis is performed by solving the coupled Navier-Stokes and heat conduction equations with heat flux transport at the fluid/solid interfaces. This will prove to be useful in allowing designers to access state-of-the-art multidisciplinary tools which will help reduce design cycle time for NPSS components and will shorten the design time for the entire engine.

DUO-MIN LIN

Research Area: Turbulence Modeling of Swirling Flows

Fluid swirling flows are not only of considerable theoretical interest, but also of great practical importance. An elementary and common rotating configuration is that of flow through a rotating pipe (see Reich and Beer 1989). Another rather elementary and common rotating configuration is the case of turbulent mixing of swirling coaxial jets discharging into an expanded duct (Roback and Johnson 1983). It is well-known that current turbulence models do not always give satisfactory results for strong-swirling flows in which eddies and vortex stretching are common features. Over the past few years, modeling of turbulent flows in rotating systems have received considerable attention.

The purpose of this study is to make some improvements to the Reynolds stress transport equation models, which are the most complicated of the turbulence models. In this application to numerical computing, strong-swirling turbulent flows are considered in two elementary rotating configurations.

The first step of the research was to study a wide variety of existing Reynolds stress turbulence models. The IP Reynolds stress turbulence transport model was selected from those models and was expressed in the general tensor form in a cylindrical system for numerical computations.

The second step of the study was to consider the existing FAST2d FORTRAN code, which has been successfully used here, computing numerous forms of turbulent flows with Reynolds stress algebraic turbulence models. The FAST2D computing algorithm with a structured grid generation technique was successfully applied to numerically compute the Reynolds stress transport turbulence models.

The third step was to improve the IP turbulence model by numerical computations and to perform a comparison of results between the $k-\epsilon$ models and the IP model (or the corresponding modified models) for the two basic rotating flows.

The last step was to complete the computational simulations of strong-swirling turbulent flows with the Reynolds stress transport turbulence models and with an improved second-moment closure turbulence model.

The study is now in the third step and good numerical results are expected. The concluding portion of the investigation will be completed at Caltech.

The current status of this work is as follows:

- (i) The IP Reynolds stress transport model equation has been written in tensor form for cylindrical coordinate systems;
- (ii) The FAST2D algorithm has been successfully used to numerically compute the Reynolds stress transport equation model;
- (iii) This effort is a good example for solving other Reynolds stress transport equation models with more closure equations, such as the three-vortex second-moment closure turbulence model.

WILLIAM LIOU

Research Area: Turbulence Modelling

The low pressure (LP) turbine is typically the power device in a turboshaft engine. A high efficiency LP turbine has become increasingly important as the new generation turbofan engines demand more power from the LP turbine than ever before. Improved performance also has to be achieved at reduced weight and cost. The efficiency of the LP turbine varies in different aircraft operating modes. For instance, at cruise, the LP turbine loses 2% in efficiency compared with that in takeoff mode. Therefore, reducing the loss of LP turbine efficiency at various points in the flight envelope is of primary concern.

In order to improve and optimize the LP turbine efficiency at all aircraft operating conditions, it is important for a turbine designer to be able to predict the LP turbine flow computationally. An accurate modeling of the flow mechanism may also lead to a reduced design cost for the LP turbine.

The work performed during 1996 was focused on modifying the ALLSPD-3D code for the model development and the calculations of flows in LP turbine passages.

A CMOTT transition model based on the shape factor was implemented into the ALLSPD-3D code. The results showed that the model successfully reduced the eddy viscosity level in the zone of flow transition. The calculations using this model were difficult to converge. Further refinement of the model may be needed.

An elliptic surface grid generation scheme was applied successfully to generate the o-mesh surrounding the turbine blade for the CHIMERA grids. A high order time discretization is also being incorporated into the ALLSPD-3D code.

The ALLSPD-3D code was applied to calculate the experimental flow from University of Minnesota. The results showed that for a low Reynolds number, the CMOTT two-equation model can successfully capture the essential features of a separated flow transition observed in the experiment. The future goal will be to achieve quantitative agreement between the model prediction and the experiment data.

JAMES LOELLBACH

Research Area: 3D Structured Grid Generation Codes for Turbomachinery

This research is focused primarily on the generation of computational grids in support of numerical flow analyses of turbomachinery components. Over the past several years, a set of programs have been developed to simplify the generation of structured grids for specific turbomachinery configurations. The grid topology consists of a non-periodic H-mesh centered around a single blade in a blade row. Both axial and centrifugal machines can be treated. The design goals for the programs are: 1) to be applicable to a sufficiently wide range of configurations with a minimum of input variables, 2) to run quickly and reliably on a wide variety of computer platforms, and 3) to produce grids of sufficiently high quality to be used with a variety of turbomachinery flow solvers. The programs utilize an algebraic transfinite interpolation method, with additional algebraic or elliptic smoothing when necessary. The required input data consists of geometry descriptions for the blade and the domain boundaries, and a small set of parameters which control the dimensions of the grid and the clustering of grid points near solid walls. Arbitrary shapes can be specified for the hub, casing, inlet and exit.

During the past year, effort has been concentrated on extending the capabilities of the grid generation codes and on applying them to a variety of turbomachinery flow problems. The primary extensions to the codes were to add the ability to accurately represent blade tip clearances and to stack multiple blade rows for stage computations. Applications included flow analyses of a wide variety of turbomachinery problems including subsonic and transonic axial compressors and turbines, axial turbine stages, centrifugal compressors, and incompressible flow through a propulsion pump.

In addition to the above grid generation effort, James Loellbach cooperated on the development of an unstructured-mesh flow solver for turbomachinery flows. This code development has been carried out primarily by Fu-Lin Tsung of ICOMP. His involvement so far has been limited to applications for testing and validation purposes. James Loellbach has generated grids for a variety of configurations and applied the flow solver to a turbine secondary flow passage and a centrifugal compressor stage.

R. R. MANKBADI

Research Area: Aeroacoustics

A Surface Integral Technique for Extension of Near Field Jet Noise Computation to the Far Field

While large-scale simulation of jet noise is the most thorough technique currently available for jet noise prediction, three-dimensional direct computation of both the near and far field requires prohibitive computer capabilities. To overcome this difficulty, a surface-integral formulation is developed which calculates the far-field sound by using the computed pressure field over a cylindrical surface surrounding the nonlinear sound source. Unlike the conventional Kirchhoff's approach, this new Surface-Integral-Formulation (SIF) does not require the normal derivative of the pressure which is a source of numerical error. The method is found to be as accurate as direct large-scale simulations, with the advantage that the use of the new method reduces the CPU time by an order of magnitude.

ANDREW T. NORRIS*Research Area: Aerothermochemistry***1. Motivation and Objective**

The objective of this work is the development, implementation and validation of thermochemical models for use in combustion codes. The principal goal in this work was to achieve significant reduction in the CPU time required for reacting flow calculations, without sacrificing accuracy. In addition, some further simulations of reacting flows were performed using the Lewis Probability Density Function (PDF) solver. This was reported at the Joint Propulsion Conference in Florida, July 6-9, 1996.

2. Work Performed**2.1 Manifold Reaction Schemes.**

During the year, the Intrinsic Low-Dimensional Manifold code was developed. This code takes a full reaction mechanism, and automatically simplifies it into a reduced mechanism. The technique has the advantage of requiring no knowledge of detailed combustion chemistry, and also shows significant improvement in accuracy over existing reduced mechanisms. For example, the popular one-step schemes of Westbrook and Dryer are similar in complexity to the ILDM method, but over-predict adiabatic flame temperature by 100-200K and only contain information about the major species. The ILDM scheme, however, contains information about all the major and minor species, and so predicts the adiabatic flame temperature accurately.

Validation tests showed the accuracy of the ILDM scheme, parameterized by two scalars, to be of the same order of accuracy as a reduced mechanism containing 12 species and 10 rate equations for PSR calculations.

The ILDM mechanisms are currently being applied to the National Combustor Code (NCC) project.

2.2 NOx Post-Processor

The NOx Post processor, based on the ILDM reduced reaction scheme, was also worked on. Its capabilities were upgraded to include JetA as a fuel and is currently being used by Pratt and Whitney for pollutant prediction in their design codes.

2.3 PDF Code

A calculation was performed using the LPDF2D code with improved CMOTT turbulence models to predict the flow in a can combustor. This work tested the turbulence models, and also the new convection schemes developed by for the PDF code. Results showed that with the improvements, the PDF part of the code only took 40% of the total CPU time required. This work was reported at the 1996 Joint Propulsion Conference.

3. Continuing Work.

During the next year, the following projects will be pursued:

1. Integration of the ILDM scheme into the NCC code.
2. Extend ILDM scheme to include another parameterizing scalar.
3. Extension of NOx post processor to include soot formation.

In addition, details of the ILDM code and the NOx post processor will be reported at the Joint Propulsion Conference in July 1997.

ALEXANDER ORON

Research Area:

This work, in collaboration with Dr. J. C. Duh of NASA, was focused on the nonlinear stability analysis of a free liquid film subjected to the Marangoni instability, in continuation of the effort started during Dr. Oron's visits to ICOMP in 1992/93 when linear stability analysis of the problem was carried out^{1,2}. A set of nonlinear evolution equations describing the spatio-temporal behavior of the system was derived and preliminary numerical solutions of these equations were obtained. For low values of the Marangoni number the liquid film is shown to rupture, while the value of its minimal thickness accelerates toward zero. However, for sufficiently large Marangoni numbers the minimal film thickness decelerates while approaching zero. This suggests a qualitative similarity between the theoretical results and the experiments on suppression of bubble coalescence by shear and temperature gradients reported recently by Dell'Aversana, Banavar and Koplik³. More work is needed to investigate the behavior of the film at small thicknesses in order to understand whether the film ruptures in a finite time or not.

References

- [1] A. Oron, R. J. Deissler, and J. C. Duh, *Eur. J. Mech. B/Fluids* **14**, 737 (1995).
- [2] A. Oron, R. J. Deissler, and J. C. Duh, *Adv. Space Res.*, 16 83 (1995).
- [3] P. Dell'Aversana, J. R. Banavar, and J. Koplik, *Phys. Fluids A* **8**, 15 (1996).

RICHARD PLETCHER

Research Area: Unstructured Grids, Large Eddy Simulation, Preconditioning

Work continued with Philip Jorgenson on the simulation of internal viscous flows using unstructured grids. Final revisions were made on a paper accepted for publication (Jorgenson, P.C.E. and Pletcher, R. H., "An Implicit Numerical Scheme for the Simulation of Internal Viscous Flows on Unstructured Grids"). Work on unstructured grid schemes for combustion applications with NASA Graduate Student Researcher Rob Cupples continued. Several sparse matrix schemes for solving the algebraic system have been evaluated. A technical paper, (Ramin, T.H. and Pletcher, R. H., "Parallel Implementation for Solving the Two-Dimensional Navier-Stokes Equations on a Cell-Centered, Unstructured Grid") based on the work of NASA graduate Student Researcher Tom Ramin has been accepted for publication.

Work was initiated with NASA Graduate Student Researcher Todd Simons to apply unstructured grid methodology to the large eddy simulation of turbulent flow. A paper (Dailey, L.D., Simons, T.A. and Pletcher, R. H. "Large Eddy Simulation of Isotropic Decaying Turbulence with Structured and Unstructured Grid Finite Volume Methods) describing some early results will be presented at the International Congress of Mechanical Engineers in November 1996. The long term objective of this research thrust is to apply LES and DNS to critical problem areas arising in propulsion systems. Such simulations can be used to enlarge the data base of knowledge needed to understand the physics of complex flows.

In the area of convergence acceleration, a paper (Dailey, L.D. and Pletcher, R. H., "Evaluation of Multigrid Acceleration for Preconditioned Time-Accurate Navier-Stokes Algorithms") utilizing the low Mach number preconditioning developed with Phil Jorgenson has been accepted for publication. The paper also evaluates the use of multigrid to accelerate convergence of an explicit Runge-Kutta scheme and an implicit LU-SGS scheme. For steady flows, the multigrid was observed to accelerate the convergence of the LU-SGS scheme for solving the compressible Navier-Stokes equations by factors ranging up to 11.5.

OLEG RYZHOV

Research Area: Boundary Layer Investigations

Dr. Ryzhov discussed nonlinear instability of a 3D boundary layer. He also discussed some problems of gas turbine flows and possible correlations between various parameters in the transition stage.

AAMIR SHABBIR

Research Area: Turbulence Modelling

The work reported here grew out of an ongoing effort to assess the performance of different turbulence models for turbomachinery flows and is being carried out in collaboration with other researchers within the Science and Technology Directorate of NASA LeRC.

In last year's annual report, Dr. Shabbir described the results of three turbulence models for a high speed axial flow compressor rotor. The models were implemented into the same CFD platform (which solves the 3D Reynolds averaged Navier-Stokes equations), and results were compared with the detailed experimental data of Suder (1994). Unlike the experiment, the CFD simulations were unable to re-produce the loss in the axisymmetric total pressure profile near the hub.

In order to sort out this discrepancy it was decided to investigate the effect of hub leakage flow, which can emanate from the small gap between the stationary and rotating parts of the hub upstream of the rotor, on the flow field near the hub. In this study this leakage flow is characterized by the following three parameters: the mass flow; the azimuthal velocity; and the shape of the azimuthal profile. It is found that the introduction of the leakage flow produces the loss seen in the experimental axisymmetric profile of total pressure. CFD results show that the leakage flow causes a flow separation in the hub-trailing edge region. The separated flow region leads to a loss in total pressure which appears as a reduction in the total pressure profile. Details of the results are given in Ref 2.

References

1. Suder, K., 1994, "Rotor 37 Blind Test Case", ASME Turbo Expo, The Hague, Netherlands.

2. A. Shabbir, M. L. Celestina, J. J. Adamczyk and A. J. Strazisar, 1997, "The Effect of Hub Leakage Flow on Two High Speed Axial Flow Compressor Rotors", To be presented at the ASME Turbo Expo, June 2-5, Orlando.

SHYUE-HORNG SHIH

Research Area: Aeroacoustics

Accurate prediction of jet noise is essential in developing advanced aircraft engines. Effort for developing noise prediction capabilities using Computational Aero-Acoustics techniques was continued.

Nonlinear effects may be important in jets. It plays a key role in the saturation of the flow disturbances representing the sound source and in the sound transmission inside the inner region. Furthermore, nonlinear interaction among the various modes of the flow disturbances could be a key noise generation mechanism. The effects of nonlinearity and of the spectra of the inflow disturbances on the radiated sound were examined using Large-Scale Simulation (LSS) approach. The nonlinear effects on the sound source in supersonic jets mimic that of subsonic jets. The initially linear streamwise growth of the disturbances is followed by nonlinear saturation. The location of the peak moves upstream with increasing the level of inflow disturbances and/or with increasing the number of modes. Because of the nonlinear mechanism, other modes are generated besides the forced ones. Subsequent nonlinear interactions control the streamwise development of the modes and therefore the radiated sound. Acoustic radiation is emitted from the streamwise location where the source peaks. The shape of the sound radiation pattern is dependent on the frequency content of the inflow disturbances. This indicates a potential for controlling jet noise by controlling the frequency content of the inflow disturbances.

During the past year, a zonal approach for prediction of jet noise was developed. A comprehensive comparison of various acoustic methodologies for far field jet noise prediction was carried out. The jet noise computation consists of two parts. LSS is used to obtain the nonlinear near field sound source. Linearized Euler Equations (LEE) and the integral methods (Lighthill, and Kirchhoff-type methods) are used to calculate the far field sound using the results obtained via LSS for the near field. A field solution is required for LEE, while the integral methods can be used to predict the noise at certain locations. Among the integral methods, a new surface integral formulation (SIF) was derived in which the calculated pressure on the cylindrical surface is used to obtain the far field sound, without the need for the normal derivative of the pressure. The LEE, SIF, Kirchhoff, and modified Kirchhoff methods provide solutions which are consistent with the solution of LSS in a supersonic jet. Discrepancy is observed when using the Lighthill's theory. The computation time is reduced substantially when the Kirchhoff-type methods are used.

This research has been conducted in close collaboration with Duane Ray Hixon of ICOMP, R. R. Mankbadi of Cairo University, Egypt (formerly, of NASA Lewis Research Center), and L. A. Povinelli of NASA Lewis Research Center as Technical Monitor. Part of the research in far field noise prediction methods was done in collaboration with J. T. Stuart of the Imperial College, UK, A. Lyrintzis of Purdue University and A. Pilon of University of Minnesota.

TSAN-HSING SHIH

Research Area: Turbulence Modelling

In FY1996, Dr. Shih's research mainly included the following: modeling of swirling flows; development of a cubic Reynolds-stress algebraic equation model; development of a pressure-strain model written in terms of second order closures and development of a compressible turbulence model. Three visitors were involved in some of these research projects.

Dr. Duo-min Lin from the California Institute of Technology worked on swirling flows using second order closures during his three month visit. The basic equations and models were formulated in a curvilinear cylindrical coordinate and partially coded into the FAST2D code.

Professor Akira Yoshizawa from the University of Tokyo worked on compressible turbulence modeling with a Markovianized two-scale method during his three weeks at ICOMP.

Professor Arne Johansson and his student Mr. Sjogren worked on high order Reynolds-stress algebraic equation model during their two weeks at ICOMP.

Very positive results were obtained from the collaborations with these three visiting researchers.

It is shown that modeling of swirling flow, which occurs in all propulsion systems, needs high order Reynolds stress algebraic equation models or second order closure models. Dr. Shih studied and proposed a cubic Reynolds stress model, which is the simplest high order algebraic equation model, yet, it has the potential for modeling swirling flows. This model is being tested and is under further development for CFD applications. Second order closure of turbulence modeling is always considered as an advanced model for complex turbulent flows with complex turbulence physics. Therefore, Dr. Shih will work on it in collaboration with Professor John L. Lumley of Cornell University. The main focus will be to model the pressure-strain terms in the second moment equations. A new model was proposed using available DNS data.

JOHN SLATER

Research Area: Computation of Unsteady Flows with Moving Geometry

Dr. Slater's research has focused on the development of computational methods for the analysis of unsteady, turbulent flows about objects with surfaces in relative motion. Last year, a dynamic planar grid generator was developed, version 2.0 of the NPARC2D flow solver to include a moving grid capability was modified, and the modified code was applied to the analysis of a variable-geometry inlet.

The primary application has been the analysis of the unstart/restart operation of a high-speed, axisymmetric inlet involving a translating and collapsing centerbody. This work contributes to the CFD/Controls activity of the Interdisciplinary Technology Office at NASA Lewis. In June of 1996 a paper entitled "Computation of Unsteady Aeropropulsion Flows with Moving Geometry" was presented at the First AFOSR Conference on Dynamic Motion CFD held at Rutgers University, New Jersey. The modifications and capabilities for a moving grid are now being implemented into version 3.0 of NPARC2D so that computations may be performed in a parallel computing environment. Plans are to extend the concepts to NPARC3D.

Other activities have involved providing small-scale support for several projects with regards to three-dimensional geometry modeling and grid generation.

The past year has included a collaboration with Don Freund and Professor Miklos Sajben of the University of Cincinnati. Dr. Slater performed some computational analyses of the rapid collapse of a flexible bump in an annular duct as part of a verification of the design of an apparatus to be used for their experiment. Don presented a paper entitled "Compressor-Face Boundary Condition Experiment: Generation of Acoustic Pulses in Annular Ducts" at the AIAA Joint Propulsion Conference in July in which he was listed as a co-author. The work has been submitted to the AIAA Journal as a test case for dynamic motion CFD. The work is also the basis of a paper that will be presented at the AIAA CFD conference this summer.

ERLENDUR STEINTHORSSON

Research Area: Code Development for Flows in Complex Geometries such as Turbine Blade Coolant Passages

This year, considerable effort was spent on validating of the TRAF3D.MB code and applying the code to study several turbomachinery flows. This work was done in cooperation with Dr. Ali Ameri, Dr. David L. Rigby, and Dr. Vijay Garg, all of whom are resident researchers at NASA Lewis Research Center. The applications included flow over a turbine blade with a squealer tip (a recessed tip) and flows in channels with ribbed walls and bleed holes (models of internal coolant passages in turbine blades). The code was also used in simulations of flow through the "branched duct" geometry, also used in earlier simulations, mainly for the purpose of evaluating the performance of the newly implemented $k-\omega$ turbulence model as compared to the conceptually simple Baldwin-Lomax turbulence model. In all cases where the code was used in simulations of flows in geometries used in experiments with well defined boundary conditions, the code was found to perform well and to produce accurate heat transfer results.

Considerable effort was spent on grid generation and related issues in the past year. Much of the effort focused on the development of multi-block grid topologies for turbine blades with tip clearance. Out of this effort came several topologies that accomplish the task of generating grid systems that are smooth and nearly orthogonal everywhere and perfectly continuous at all block boundaries, including periodic boundaries. They also eliminate the layer of densely clustered grid surfaces that in traditional structured grids emanate into the free stream region from the boundary-layer grid on the tip.

In the past year or two it has become clear that multiblock grid systems for complicated geometries must in general be created by using a large number of relatively small "elemental" blocks. However, to maximize the performance of flow solvers, it is desirable to use few and large blocks. Therefore, elemental blocks produced by the grid generator need to be merged to minimize computational expense. Through block merging, it is frequently possible to reduce the number of blocks by a factor of ten or more (for large complicated grid systems). Such merging of blocks is extremely tedious to do by hand if the number of blocks is greater than four or five. Thus, a computer program was developed for the automatic merging of blocks. The program receives as input a "schedule" of which blocks to merge and produces the merged blocks along with connectivity data. The merging schedule is generated automatically using an algorithm and a computer program developed by Dr. Rigby. Together these two computer programs make multiblock grids easy to use in computations.

Research on multi-fluid flow simulations in complex fuel atomizers, initiated last year in cooperation with Parker Hannifin Corporation, continued. The research project involved the use of a front tracking

method to model the gas-liquid interface and a projection method to simulate the incompressible flow of the gas and liquid phases. The project proved the feasibility of simulating the unsteady axisymmetric flow in simplex atomizers by using the front tracking method but also showed that improved algorithms were needed to enhance computational efficiency and robustness. In particular it was found that the Euler and Adams-Bashforth time integration schemes that were tested have unacceptably restrictive time step limitations for Reynolds numbers typical of simplex atomizers. Also, the projection method, originally implemented using an SOR solver, was found hard to use for this unsteady problem involving inflow and outflow. The former issue was addressed by introducing a four-stage Runge-Kutta time stepping scheme. This relieved the restrictive "viscous" time-step limitation of the previous time stepping scheme. The latter issue was addressed temporarily by introducing GMRES solver with ILU preconditioning for the projection step. This improved the robustness of the solver but the cost of the projection step was still high. Sample results from the simulations of the simplex nozzle flows were exhibited at the NASA Lewis Business Industry Summit. The exhibit generated several inquiries into the multi-fluid flow simulation technology. The work on multi-fluid flow simulation was done in cooperation with Dr. Ajmani of ICOMP.

GERALD TRUMMER

Area: System Administration

Mr. Trummer is performing software systems integration, systems administrative tasks and software maintenance on the Aeromechanics Division's network of workstations. This includes installing and upgrading operating systems, graphics packages, as well as making modifications to existing graphics source code as required by the needs of the scientists and researchers in the division. He is also working on integrating CFD codes developed by the researchers with graphics packages to perform real-time computation and visual analysis. He has made movies (video tape, mpeg, quicktime) of CFD work done by researchers for scientific analysis and display at remote sites by colleagues.

GRETAR TRYGGVASON

Research Area: Front Tracking Techniques, Simulations of Multifluid and Multiphase Solutions

Dr. Tryggvason continued to work on improvements and additional capabilities for an axisymmetric code for multifluid simulations. This year, he included the capability to incorporate thermal migration and insoluble contaminants into a variable grid code. This allows simulations of bubbles and drops in very large domains as well as grid refinement in critical areas such as near a wall. He also generalized a method to do "topology changes" when fluid blobs coalesce or break up, interacted with David Jacqmin on multifluid simulations, with Erlendur Steinthorsson on simulations of the generation of sprays by swirling flows, and wrote various diagnostic software.

FU-LIN TSUNG

Research Area: Development of 3D Structured/Unstructured Hybrid Navier-Stokes Solver for Turbomachinery

The goal for the present research is to develop, validate, and apply a 3-D, Navier-Stokes, unstructured-grid solver for turbomachinery applications. In 1996, new routines have been added to extend the code's capability to include unsteady interaction problems. For unsteady problems, first- and second-order time accuracy is achieved by using Newton subiteration for the temporal integration term. For interaction problems, boundary conditions are now available to handle sliding interfaces with interaction between two domains with a nonzero relative motion between the domains. The solver has

been applied to both axial and centrifugal geometries including a Lewis turbine stage with stator-rotor interaction, a centrifugal compressor with impeller and diffuser-vane interaction, and also a turbine stage coupled with a secondary rim-seal flow passage. Code validation for the new features are being performed at this time by comparing the computed results with currently, or soon to be, available experimental data.

ELI TURKEL

Research Area: Boundary Conditions, Oiler Solvers

Dr. Turkel extended preconditioning techniques from external to internal flows. This required a change of the boundary conditions imposed both at inflow and outflow. Also extension to rotors are currently being investigated. A flow around a stator at $M = 0.1$ has been successfully run. The low speed preconditioning algorithm was adapted for use in internal flows. In particular, this scheme was programmed for two different NASA Lewis codes with assistance from Mark Celestina, Rod Chima and Dan Totweed. Several tests on pumps, impellers and cascades gave encouraging results.

JIE WU

Research Area: Development of Finite Element Methods for Fluid Mechanics and Electromagnetics

Dr. Wu worked on numerical simulations of evolving interfaces based on a continuum approach. The problems studied include both two-fluid flows and crystal growth. In both cases surface tension is an important factor in the system. In the continuum approach the different phases are identified through the use of a color function (or phase-field function). The interfaces between different fluids are represented as transition regions with finite thicknesses. The evolution of the interfaces is captured by solving the convective equation of the color function. The surface tension effect, proportional to the curvature of the interface, is presented in terms of stresses. The stresses are defined by the gradient of the color function. The finite element method is used to discretize the equation system.

1. Two-Fluid Flows

In this problem standard incompressible Navier-Stokes equations with variable material property (density and viscosity) are solved. In addition, a convective equation for the color function is solved. The surface tension effect comes as source terms in the momentum equations. A number of numerical tests have been carried out for two-dimensional and axis-symmetric cases and the results are compared with analytical, experimental or other numerical data when available. Good agreement has been obtained.

2. Crystal Growth

Crystal growth in saturated liquid was studied based on the continuum approach using the finite element method. A heat conduction equation and a phase field equation are solved. The whole computational domain is treated equally and the location of the interface is not explicitly required. The surface tension affects the velocity at which the interface advances. Four and six-fold anisotropy was considered. A large number of numerical tests have been carried out involving different values of undercooling, surface tension and strength of anisotropy. Dendritic growth patterns have been obtained which shows both the stable growth of the major branches at steady speed and unstable growth of side branches. A good agreement with other results in literature has been obtained.

XUESONG WU*Research Area:*

Dr. Wu had frequent discussions with M. E. Goldstein, L. Hultgren, S. J. Leib and S. S. Lee. They discussed various issues regarding recent theoretical development of Laminar-Turbulent transition research. Such an exchange of ideas is beneficial to both NASA and Dr. Wu. He also worked with S. J. Leib and M. E. Goldstein on interactions between pairs of Tollmien-Schlichting waves. They are currently producing a co-authored paper.

AKIRA YOSHIZAWA*Research Area:*

Dr. Yoshizawa conducted research in the development of a compressible turbulence model applicable to high-speed flows. This is based on incorporating effects of compressibility into the turbulent viscosity. Dr. Yoshizawa succeeded in capturing the characteristics of high-speed flows, such as the steep decrease in the growth rate of a shear layer.

SHAYE YUNGSTER*Research Area: High Speed Combustion and Detonation Waves*

The research work for the 1996 year focused on three different areas:

1) CFD Analysis of Rocket-Based Combined Cycle (RBCC) propulsion system

In recent years, rocket-based combined-cycle (RBCC) engines have been identified as potential propulsion candidates for single-stage to orbit and hypersonic cruise vehicles. RBCC engines combine the high thrust-to-weight ratio of rockets with the high specific impulse of ramjets into a single, integrated propulsion system. Several RBCC engine concepts are currently being investigated at the NASA Lewis Research Center. Experiments have been carried out at the NASA Lewis' Hypersonic Tunnel Facility (HTF) and 1'x1' supersonic wind tunnel. In support of this experimental work, computational fluid dynamic (CFD) tools are being developed for analyzing RBCC flowfields.

The objective of this work was to evaluate the CFD approach by computing the flowfield through a subscale RBCC engine inlet for Mach numbers of 5 and 6. The computations were based on the Navier-Stokes solver NPARC, and results were compared with the experimental data obtained in the NASA Lewis 1'x1' supersonic wind tunnel. To simulate the effects of combustion on inlet performance, a back-pressure was imposed at the engine exit plane. The study investigated a wide range of back-pressures, from super-critical operation (i.e., no back-pressure) to critical condition (i.e., maximum back pressure before inlet unstart). The computed static and pitot pressure profiles were in good agreement with experimental data. The numerical simulations provided details of the flowfield not obtained in the experiments, and will provide guidance for the design of improved inlets and, by analyzing the flow distribution at the base of the struts, help determine the optimum location for the fuel injectors. The results of this work, in collaboration with J.R. DeBonis, were presented in ICOMP-96-6 (also AIAA paper 96-3145 and NASA TM 107274) and in AIAA paper 97-0028.

2) Development of Global RBCC Engine Cycle Code

In addition to the experimental and CFD efforts described previously, research work at the NASA Lewis Research Center is also focusing on the development of a global RBCC engine cycle code for performance analysis of the various propulsion modes. As part of this effort, a computer program for analyzing the ejector/mixer component of the engine was developed using a one-dimensional approach based on the solution of the continuity, momentum, and energy conservation equations applied to a control volume around the ejector/mixer. The analysis assumes steady, piecewise uniform flow of thermally perfect gases for the primary and secondary streams at the inflow plane and uniform flow in chemical equilibrium at the exit plane. (The equilibrium computations are based on the CSDTTP chemical equilibrium program).

For a given configuration and specified primary and secondary flow parameters, the program determines whether the ejector/mixer is thermally choked, aerodynamically choked, or not choked at all, and computes the solution accordingly. Combustion efficiency, and viscous and heat transfer effects can be included.

A copy of this program has been sent to Pratt & Whitney per their request.

3) Detonation Wave Modeling

The development of several new propulsion systems such as the ram accelerator, the oblique detonation wave engine and the pulse detonation engine, all of which utilize detonation waves in their thermodynamic cycle, has stimulated interest in this mode of combustion. In this work (done in collaboration with Krishnan Radhakrishnan) the temporal evolution of planar detonation waves in a shock tube were simulated. The one-dimensional equations for chemically reacting flow were solved in a fully coupled manner, using an implicit, time accurate algorithm. The solution procedure is based on a spatially second order total variation diminishing scheme and a temporally second order, variable-step, backward differentiation formula method. Calculations were carried out for three different gas mixtures of hydrogen oxygen and argon, and the results compared with calculations obtained with a chemical equilibrium code. The hydrogen-oxygen chemistry was modeled with a 9-species, 19-step mechanism. The computed steady-state detonation speeds were in very good agreement with those predicted by the chemical equilibrium code. The eventual goal of our modeling effort is to simulate nonplanar multidimensional detonation, including viscous effects, in order to improve our understanding of the operation and limitations of detonation-wave-based propulsion devices. Results of this work were presented in an AIAA paper 96-2949.

JIANG ZHU

Research Area: Turbulence Modelling

The CMOTT turbulence module, version 2.0, has been developed for the NPARC code. The module is written in a self-contained manner so that the user can use any turbulence model in the module without being concerned about how to implement or solve it. Three two-equation turbulence models have been built into the module: Chien, Shih-Lumley and CMOTT models, and all of them have both the low Reynolds number and wall function options. Unlike Chien's model, both the Shih-Lumley and CMOTT models do not involve the dimensionless wall distance y^+ in the low Reynolds number approach, an advantage for separated flow calculations. The Van Driest transformation is used so that the wall functions can be applied to both incompressible and compressible flows. The module has been fully

validated against two test problems, one for subsonic and the other for transonic flow. The module can be easily linked to the NPARC code for practical applications.

Reports and Abstracts

1996 REPORTS AND ABSTRACTS

Keith, Theo G., Balog, Karen, and Povinelli, Louis A., Editors: "Institute for Computational Mechanics in Propulsion (ICOMP), Tenth Annual Report - 1995", ICOMP Report 96-1, NASA TM 107248, June, 1996, 63 pages.

The Institute for Computational Mechanics in Propulsion (ICOMP) is operated by the Ohio Aerospace Institute (OAI) and funded under a cooperative agreement by the NASA Lewis Research Center in Cleveland, Ohio. The purpose of ICOMP is to develop techniques to improve problem-solving capabilities in all aspects of computational mechanics related to propulsion. This report describes the activities and accomplishments during 1995.

Liou, William W., and Shih, Tsan-Hsing (ICOMP): "Transonic Turbulent Flow Predictions With New Two-Equation Turbulence Models", ICOMP Report 96-2, CMOTT 96-01, NASA CR 198444, January, 1996, 12 pages.

Solutions of the Favre-averaged Navier-Stokes equations for two well-documented transonic turbulent flows are compared in detail with existing experimental data in this paper. While the boundary layer in the first case remains attached, a region of extensive flow separation has been observed in the second case. Two recently developed $k-\epsilon$, two-equation, eddy-viscosity models are used to model the turbulence field. These models satisfy the realizability constraints of the Reynolds stresses. Comparisons with the measurements are made for the wall pressure distribution, the mean streamwise velocity profiles and turbulent quantities. Reasonably good agreement is obtained with the experimental data.

Liou, William W. (ICOMP); and Huang, P. G. (MCAT, Inc.): "Calculations of Oblique Shock Wave/Turbulent Boundary-Layer Interactions With New Two-Equation Turbulence Models", ICOMP Report 96-3, CMOTT 96-2, NASA CR 198445, January, 1996, 10 pages.

Supersonic flows involving oblique shock wave/turbulent boundary-layer interactions are studied using the Favre-averaged Navier-Stokes equations and two recently developed $k-\epsilon$, two-equation, eddy-viscosity models. The primary difference between these models and the existing $k-\epsilon$ model is that the new models satisfy the realizability constraints of the Reynolds stresses. Three cases with different levels of shock strength were calculated. The corresponding flows were observed to be attached, near incipient separation, and with large separation zone, respectively. The computed results are compared with surface measurements for all the cases and, for the last case, where there is a large region of flow separation, measured mean and turbulent kinetic energy profiles are also available for comparison. The results show reasonable agreement with the measurements.

Shih, Tsan-Hsing (ICOMP): "Developments in Computational Modeling of Turbulent Flows", ICOMP Report 96-4, CMOTT 96-3, NASA CR 198458, February, 1996, 31 pages.

In this paper, we will discuss some recent model developments in two turbulence closure schemes. One is the Reynolds-stress algebraic equation model and the other is the Reynolds-stress transport equation model. Various model constraints required by the rapid distortion theory, the invariant theory and the realizability principle, etc. will be described in the model development.

REPORTS AND ABSTRACTS

Chung, Joongkee (ICOMP): "Comparison of Compressor Face Boundary Conditions for Unsteady CFD Simulations of Supersonic Inlets", ICOMP Report 96-5, NASA TM 107194, March, 1996, 9 pages.

A new type of compressor face boundary condition (BC) for an inlet section exit was developed and implemented in the NPARC code. Time-accurate computations were performed to study inlet transients caused by upstream and downstream disturbances. In general, static pressure time histories calculated at various locations are less oscillatory when compared to results for other conventionally used fixed-pressure boundary conditions. This suggests that the new BC is less reflective than the others. In particular, pressure responses to steps in compressor face Mach number show no overshoot with the new BC, which is in good qualitative agreement with experimental data. 1-D unsteady inviscid computational results calculated by the LAPIN code were in good agreement with those obtained by 2-D and 3-D NPARC codes when applied to the NASA 55-45 mixed-compression and variable diameter centerbody (VDC) inlets. For 2-D calculations, the existing Runge-Kutta explicit scheme and a newly modified implicit scheme which uses a subiteration technique to produce 2nd-order time accuracy were studied and compared. The subiteration technique allowed larger time steps with a corresponding reduction of CPU time.

DeBonis, J. R. (NASA Lewis); Yungster, S. (ICOMP): "Rocket-Based Combined Cycle Engine Technology Development-Inlet CFD Validation and Application", ICOMP Report 96-6, AIAA-96-3145, NASA TM 107274, June, 1996, 20 pages.

A CFD methodology has been developed for inlet analyses of Rocket-Based Combined Cycle (RBCC) Engines. A full Navier-Stokes analysis code, NPARC, was used in conjunction with pre- and post-processing tools to obtain a complete description of the flow field and integrated inlet performance. This methodology was developed and validated using results from a subscale test of the inlet to a RBCC "Strut-Jet" engine performed in the NASA Lewis 1x1 ft. supersonic wind tunnel. Results obtained from this study include analyses at flight Mach numbers of 5 and 6 for super-critical operation conditions. These results showed excellent agreement with experimental data. The analysis tools were also used to obtain pre-test performance and operability predictions for the RBCC demonstrator engine planned for testing in the NASA Lewis Hypersonic Test Facility. This analysis calculated the baseline fuel-off internal force of the engine which is needed to determine the net thrust with fuel on.

Steinthorsson, Erlendur (ICOMP); Ameri, Ali A. (AYT Corporation); and Rigby, David L. (NYMA, Inc.): "Simulations of Turbine Cooling Flows Using a Multiblock-Multigrid Scheme", ICOMP Report 96-7, NASA CR 198539, AIAA 96-0621, October, 1996, 15 pages.

Results from numerical simulations of air flow and heat transfer in a "branched duct" geometry are presented. The geometry contains features, including pins and a partition, as are found in coolant passages of turbine blades. The simulations were performed using a multi-block structured grid system and a finite volume discretization of the governing equations (the compressible Navier-Stokes equations). The effects of turbulence on the mean flow and heat transfer were modeled using the Baldwin-Lomax turbulence model. The computed results are compared to experimental data. It was found that the extent of some regions of high heat transfer was somewhat under predicted. It is conjectured that the underlying reason is the local nature of the turbulence model which cannot account for upstream influence on the turbulence field. In general, however, the comparison with the experimental data is favorable.

Rigby, David (NYMA, Inc.); Ameri, Ali A. (AYT Corporation); and Steinthorsson, Erlendur (ICOMP): "Internal Passage Heat Transfer Prediction Using Multiblock Grids and a $k-\omega$ Turbulence Model", ICOMP Report 96-8, NASA CR 198540, 96-GT-188, October, 1996, 12 pages.

Numerical simulations of the three-dimensional flow and heat transfer in a rectangular duct with a 180° bend were performed. Results are presented for Reynolds numbers of 17,000 and 37,000 and for aspect ratios of 0.5 and 1.0. A $k-\omega$ turbulence model with no reference to distance to a wall is used. Direct comparison between single block and multiblock grid calculations are made. Heat transfer and velocity distributions are compared to available literature with a good agreement. The multi-block grid system is seen to produce more accurate results compared to a single-block grid with the same number of cells.

Ameri, Ali A. (AYT Corporation); and Steinthorsson, E. (ICOMP): "Analysis of Gas Turbine Rotor Blade Tip and Shroud Heat Transfer", ICOMP Report 96-9, NASA CR 198541, 96-GT-189, October, 1996, 10 pages.

Predictions of the rate of heat transfer to the tip and shroud of a gas turbine rotor blade are presented. The simulations are performed with a multiblock computer code which solves the Reynolds Averaged Navier-Stokes equations. The effect of inlet boundary layer thickness as well as rotation rate on the tip and shroud heat transfer is examined. The predictions of the blade tip and shroud heat transfer are in reasonable agreement with the experimental measurements. Areas of large heat transfer rates are identified and physical reasoning for the phenomena presented.

Ameri, A. A. (AYT Corporation); and Steinthorsson, E. (ICOMP): "Prediction of Unshrouded Rotor Blade Tip Heat Transfer", ICOMP Report 96-10, NASA CR 198542, 95-GT-142, October, 1996, 11 pages.

The rate of heat transfer on the tip of a turbine rotor blade and on the blade surface in the vicinity of the tip, was successfully predicted. The computations were performed with a multiblock computer code which solves the Reynolds Averaged Navier-Stokes equations using an efficient multigrid method. The case considered for the present calculations was the SSME (Space Shuttle Main Engine) high pressure fuel side turbine. The predictions of the blade tip heat transfer agreed reasonably well with the experimental measurements using the present level of grid refinement. On the tip surface, regions with high rate of heat transfer was found to exist close to the pressure side and suction side edges. Enhancement of the heat transfer was also observed on the blade surface near the tip. Further comparison of the predictions was performed with results obtained from correlations based on fully developed channel flow.

Hixon, R. (ICOMP): "On Increasing the Accuracy of MacCormack Schemes for Aeroacoustic Applications", ICOMP Report 96-11, NASA CR 202311, December, 1996, 30 pages.

Due to their inherent dissipation and stability, the MacCormack scheme and its variants have been widely used in the computation of unsteady flow and acoustic problems. However, these schemes require many points per wavelength in order to propagate waves with a reasonable amount of accuracy. In this work, the linear wave propagation characteristics of MacCormack-type schemes are investigated, and methods for greatly improving their performance are described and demonstrated.

Wu, Jie and Jiang, Bo-nan (ICOMP): "A Least-Squares Finite Element Method for Electromagnetic Scattering Problems", ICOMP Report 96-12, NASA CR 202313, December, 1996, 35 pages.

The least-squares finite element method (LSFEM) is applied to electromagnetic scattering and radar cross section (RCS) calculations. In contrast to most existing numerical approaches, in which divergence-free constraints are omitted, the LSFEM directly incorporates two divergence equations in the discretization process. The importance of including the divergence equations is demonstrated by showing that otherwise spurious solutions with large divergence occur near the scatterers. The LSFEM is based on unstructured grids and possesses full flexibility in handling complex geometry and local refinement. Moreover, the LSFEM does not require any special handling, such as upwinding, staggered grids, artificial dissipation, flux-differencing, etc. Implicit time discretization is used and the scheme is unconditionally stable. By using a matrix-free iterative method, the computational cost and memory requirement for the present scheme is competitive with other approaches. The accuracy of the LSFEM is verified by several benchmark test problems.

Wu, Jie (ICOMP); Yu, Sheng-Tao (NYMA) and Jiang, Bo-nan (ICOMP): "Simulation of Two-Fluid Flows by the Least-Squares Finite Element Method Using a Continuum Surface Tension Model", ICOMP Report 96-13, NASA CR 202314, December, 1996, 26 pages.

In this paper a numerical procedure for simulating two-fluid flows is presented. This procedure is based on the Volume of Fluid (VOF) method proposed by Hirt and Nichols and the continuum surface force (CSF) model developed by Brackbill, et al. In the VOF method fluids of different properties are identified through the use of a continuous field variable (color function). The color function assigns a unique constant (color) to each fluid. The interfaces between different fluids are distinct due to sharp gradients of the color function. The evolution of the interfaces is captured by solving the convective equation of the color function. The CSF model is used as a means to treat surface tension effect at the interfaces. Here a modified version of the CSF model, proposed by Jacqmin, is used to calculate the tension force. In the modified version, the force term is obtained by calculating the divergence of a stress tensor defined by the gradient of the color function. In its analytical form, this stress formulation is equivalent to the original CSF model. Numerically, however, the use of the stress formulation has some advantages over the original CSF model, as it bypasses the difficulty in approximating the curvatures of the interfaces. The least-squares finite element method (LSFEM) is used to discretize the governing equation systems. The LSFEM has proven to be effective in solving incompressible Navier-Stokes equations and pure convection equations, making it an ideal candidate for the present applications. The LSFEM handles all the equations in a unified manner without any additional special treatment such as upwinding or artificial dissipation. Various bench mark tests have been carried out for both two dimensional planar and axisymmetric flows, including a dam breaking, oscillating and stationary bubbles and a conical liquid sheet in a pressure swirl atomizer.

Seminars

1996 SEMINARS**Wolff, Mitch (Wright State University): "Nonlinear Zonal Analysis of Unsteady Compressor Aerodynamics"**

Unsteady aerodynamic phenomena continue to produce serious aeroelastic problems in the development of new turbomachinery. To minimize this development problem, accurate unsteady aerodynamic cascade models are required. Such models are generally time linearized. However, these linearized flow models may not be valid for many situations. For example, nonlinear effects are quite likely to be associated with larger amplitudes of blade oscillation as well as unsteady transonic flows with shock motion induced by oscillating blades at small amplitudes. To analyze nonlinear flow fields, unsteady Euler and Reynolds-averaged Navier-Stokes codes have been developed for blade row interactive flows and for oscillating cascade flows. Using these methods, the feasibility of analyzing complex two and three dimensional flows in turbomachinery has been demonstrated. However, they require much more computer time and storage than do linearized codes. This presentation will give a summary of analytical work to improve the efficiency of current methods by utilizing a zonal analysis. In particular, a computationally efficient zonal inviscid-viscous interaction model is developed. The model incorporates an inverse integral boundary layer solution with the time-marching Euler analysis, NPHASE, implemented on a deforming computational grid. Results from an investigation into the significant effect of viscosity on the unsteady flow will be presented. The analysis of oscillating cascades over a range of steady operating conditions clearly demonstrates the importance of considering viscous interactions. The nonlinearities of the unsteady cascade aerodynamics are increased by viscous effects. The inverse integral boundary layer model incorporated in a quasi-steady manner with a time marching Euler solution is an effective way to analyze the viscous unsteady aerodynamics of a cascade of oscillating airfoils.

Buelow, Philip E.O. (The Pennsylvania State University): "Convergence Enhancement of Euler and Navier-Stokes Algorithms"

Several issues are commonly known to degrade the performance of CFD algorithms. Among these are stiffness due to high aspect-ratio grid cells, low Mach numbers and low Reynolds numbers. The time-derivative preconditioning techniques which have been developed by Dr. Merkle and co-workers are employed to accelerate convergence by alleviating the low Mach number and low Reynolds number stiffness. Vector stability analysis is used as a guide to determine methods for improved convergence for high aspect ratio grid cells, as well as for low Mach number and low Reynolds number conditions. Initial computations of inviscid and fully-developed channel flows are performed to test the high grid aspect ratio enhancements to various algorithms. Further computations of nozzles, boundary layers, shear layers, back-step flows, and shock-boundary layer interactions show the extension to more practical flows.

The performance of several 2-D algorithms is examined, including the central-difference and upwind ADI, point Gauss-Seidel, line Gauss-Seidel, and four-stage Runge-Kutta schemes. Some issues of improved performance in 3-D are also addressed.

Mayer, Ernst (Case Western Reserve University): "Transient Disturbance Growth in Hydrodynamic Stability: Recent Progress, Unresolved Issues, and Where Next for the Field"

In the last few years, we have seen a new paradigm emerge in hydrodynamic stability, namely that in many fluid flows, even in those which have no exponentially growing modes according to classical linear stability theory, significant non-modal (or *transient*) disturbance growth may occur if the underlying differential stability operator is non-normal, i.e. has nonorthogonal eigenfunctions. There is a growing body of evidence that such non-modal growth is an important component of so-called bypass transition in channel and boundary layer flows, as well as in the short-time behavior of disturbances on flows which *do* have classically unstable eigenmodes.

In this talk, I will attempt to survey what has been accomplished so far in this area in terms of theory, computation, and comparison with experiment, and will discuss recent developments and areas in need of further study, including the following:

- 1) Non-modal analysis for compressible flows;
- 2) The need for a consistent spatial-stability formulation of the non-modal analysis;
- 3) Transient-growth analysis of unbounded flows, e.g., boundary layers, especially the role of continuous spectrum (improper eigenfunctions);
- 4) Linear vs. Nonlinear mechanisms: what's the latest score?

Nerurkar, Ashay (Research Assistant, Syracuse University): "Design Study of Turbomachine Blades by Optimization and Inverse Techniques"

A design method based on parametric optimization techniques coupled directly with a CFD analysis code for turbomachine blades will be presented. The blade geometry is parameterized and the optimization method is used to search for a blade geometry that will minimize a design objective function. The design flow criteria employed in this study include matching a prescribed swirl schedule in the bladed region (unconstrained optimization) and minimizing the stagnation pressure loss due to shock waves for a prescribed overall change in swirl across the blade row (constrained optimization). The method is demonstrated for two-dimensional cascaded blades in the transonic and supersonic flow regimes for inviscid flows. A comparative study of the blades designed by the optimization technique and an existing inverse method is presented.

Benetschik, Hannes (Institute for Computational Mechanics in Propulsion (ICOMP)): "Analysis of Two- and Three-Dimensional Turbomachinery Flows Using an Implicit Upwind Algorithm with Transition Modeling"

In this three-part presentation, the development of a computer code for the numerical simulation of turbomachinery flows will be discussed. In the first part, the basic approach used in the numerical simulations will be presented. The governing equations, including two turbulence transport modeling equations, will be discussed and their representation in vector notation as well as in different sets of coordinates required for turbomachinery flow analyses will be considered. Details of the numerical algorithm will be shown, and the motivation for upwind differencing as well as the use of the Riemann problem in a discrete solution scheme are explained. Particular emphasis will be given to the

actual approximation of the Riemann problem as well as to special modifications introduced to the well known discrete approximations to convection and diffusion terms. Subsequently, the use of implicit time integration is highlighted, where the treatment of boundary conditions will be explained. The second part of the presentation will be dedicated to engineering applications of the algorithm. Several selected turbine and compressor test cases will be used to document the current status of the computer code being developed. The final part of the presentation will focus on an approach to model transition in turbomachinery. The basic concept will be addressed and two-dimensional applications are discussed. In a final note, the prospects of the current code development are summarized.

Chang, Sin-Chung (NASA Lewis Research Center); Yu, Sheng-Tao (NYMA Technology Inc.): "The Conceptual Difference between the CE/SE Method and the Modern Upwind Schemes"

A new framework for the numerical solution of conservation laws, namely, the Method of Space-Time Conservation Element and Solution Element, or the CE/SE method for short, will be reviewed. The method is distinguished by the simplicity of its conceptual basis -- flux conservation in space and time. In describing the present method, the original papers upon which it is based will be briefly discussed. The original approach, started from basic integral equations. The algebraic details and mathematical analyses were described in the papers presented in a systematic way, such that all information needed to implement a computer program was provided. That approach made it clear that the present method was developed from fundamental physical postulates. It is not an incremental improvement of a previously existing method.

The CE/SE method, however, can also be described in an alternative and more intuitive manner by focusing on its unique space-time discretization. In particular, the essence of the method may be grasped from a simple delineation of the inherent space-time geometry. With it, the rigorous algebraic details and mathematical proofs, though not superfluous, take on more of the flavor of confirming the intuitively obvious facts. In addition, we wish to take a somewhat comparative approach to stimulate the interest among researchers familiar with modern upwind schemes. Thus, we shall place the present method into the context of the projection-evolution approach advocated by van Leer. In contrast to modern upwind schemes, the approach here does not use the Riemann solver and the reconstruction procedure as building blocks. Therefore, the logic and rationale are considerably simpler. Moreover, no directional splitting is employed and therefore the numerical resolution of two-dimensional calculations is comparable to that of one-dimensional calculations. The present approach has yielded high-resolution simulations of shocks, detonation waves, acoustic waves, vortices, and interactions among shocks, acoustic waves and vortices.

Shih, Tom I-P. (Carnegie Mellon University): "Computations of Multiphase Flows"

Many flows of practical importance involve multiple phases in which particles (liquids or solids) are dispersed in a fluid (gas or liquid). Examples include pneumatic conveying of particulate solids in transport lines, fluidized-bed combustors, and sprays. Mathematically, the most rigorous way to analyze a multiphase flow is to resolve all phases as well as all interfaces between phases. Though this approach has the potential to reveal fundamental physics, it is not practical for engineering problems. For engineering problems, what takes place at interfaces need to be modeled to reduce computational cost. This talk addresses some of the issues associated with computing multiphase flows by using the Eulerian-Eulerian approach in which both the particle and fluid phases are treated as continua. The focus will be on needs in modeling and numerical methods of solution. One method with potential to be more efficient than what is currently being used is presented. This method is demonstrated for a particle-laden flow involving solid particles dispersed in an incompressible fluid. Also, presented are results

from a few direct simulations that do resolve the particle/fluid interfaces with the goal aimed at developing better models for the Eulerian-Eulerian approach.

Dudek, Scott (University of California at Berkeley): "A Structured Grid Adaptive Mesh Refinement Multigrid Algorithm for Steady State Flows"

We present an algorithm to solve the two-dimensional Euler equations for steady-state compressible flows using structured, body-fitted grids and solution-adaptive mesh refinement. The solution is marched to steady state using an explicit, cell-centered, second-order multidimensional upwind method. Convergence is accelerated by local time stepping and a multilevel multigrid method. Logically rectangular refined meshes are embedded within regions where an estimate of the local truncation error is above a prescribed tolerance. Results are shown for subsonic and transonic internal and external flows.

The adaptive mesh refinement (AMR) algorithm allows computational resources to be utilized efficiently by placing refined grids only in areas in which they are needed. Thus, excess computations and memory are not wasted. In addition, AMR gives great flexibility in problem solving, since it is not necessary to know beforehand where high resolution will be needed. For instance, refinement around shocks is handled automatically, and shock locations do not need to be known a priori. Furthermore, since the grids are structured, and refinement is done in blocks, the data structures are simple, and optimization is straightforward, particularly on vector machines. The use of an object-oriented programming style, using C++, simplifies the implementation of AMR considerably.

Johansson, Arne (The Royal Institute of Technology): "A New Explicit Algebraic Reynolds Stress Model and Some Aspects of Model Testing Through Use of Highly Automated Code Generation"

An explicit algebraic Reynolds stress model (EARS) is presented which is based on an exact solution of the implicit equations that result from linear models of the terms in the RST-equations. Thus, no ad-hoc assumption is needed for the production to dissipation ratio. It is shown that EARS is well suited for integration to the wall and that all individual Reynolds stresses can be well predicted by introduction of only one simple wall damping function of the van Driest type. Also, some aspects of model testing through the use of automated code generation by algebra manipulation software tools are discussed.

Suresh, A. (NYMA, Inc., NASA Lewis Research Center) and Huynh, H. T. (NASA Lewis Research Center): "Accurate Monotonicity-Preserving Schemes With Runge-Kutta Time Stepping"

A new class of high-order monotonicity-preserving schemes for the numerical solution of conservation laws is presented. The interface value in these schemes is obtained by limiting a higher-order polynomial reconstruction. The limiting is designed to preserve accuracy near extrema and to work well with Runge-Kutta time stepping. Computational efficiency is enhanced by a simple test that determines whether the limiting procedure is needed. For linear advection in one dimension, these schemes are shown to be monotonicity-preserving and uniformly high-order accurate. Numerical experiments confirm that these schemes compare favorably with, and in many instances, improve upon Essentially Non-oscillatory (ENO) schemes and Weighted-ENO (WENO) schemes.

Coolant Flow Management Workshop

Coolant Flow Management Workshop December 12-13, 1996

Objective and Scope

The purpose of the Coolant Flow Workshop is to review the status of work accomplished to date on a number of research efforts and to assess the planned work effort. The coolant flow management program is a response to the 1993 Turbomachinery Peer Review committee suggestion that LeRC pursue coupling of the internal coolant passage flow and heat transfer with the external gas path through the blade/vane film cooling holes.

This program evolved through a series of meetings with designers and heat transfer researchers in industry in 1994 and 1995. Modelling, turbulence models, validation data and grid generation were identified as topics that needed to be addressed for both internal passage heat transfer and film cooling. In addition, the capability to integrate the internal and external heat transfer solutions and optimize the results was identified as a long term goal.

The Workshop will last for a day and a half and will cover three topics: Internal Coolant Passage Heat Transfer, Film Cooling, and Coolant Flow Integration and Optimization. Each session will present a combination of both analytical and experimental research and will present both current results and future research. There will be a group discussion at the end of each session for clarification of the material presented and to solicit suggestions for additional research.

This workshop is cosponsored by NASA Lewis Research Center and ICOMP, and was held at the Ohio Aerospace Institute, located adjacent to the Center.

Workshop Agenda

December 12, 1996

7:30 Registration

8:00 Welcome and Opening Remarks
Larry A. Diehl, Director of R&T

8:05 Introductory Remarks
Louis A. Povinelli, Turbomachinery and Propulsion Systems Division Chief

8:15 Overview
Joe Gladden, NASA LeRC

Session I. Internal Coolant Passage Presentations

Chair: *Ali Ameri*

8:30 On the Use of Structured Multi-Block Grids for Internal Coolant Flow Calculations
David Rigby, NYMA, Inc.

8:50 Numerical Simulation of Turbine Blade Cooling
Tom Shih, Carnegie Mellon University

9:10 Flow in Serpentine Coolant Passages with Trip Strips
David Tse, Scientific Research Assoc., Inc.

- 9:30 Experimental Heat Transfer in Turn Regions of Rotating Coolant Passages
Joel Wagner, United Technologies Research Ctr.
- 9:50 Experimental Multipass Heat Transfer with Trips & Bleed
Douglas Thurman/Philip Poinsett, NASA LeRC
- 10:10 Rotating Multipass Coolant Passages with Bleed
Sai Lau, Texas A&M
- 10:30 BREAK
- 10:50 Group Discussion
- 12:00 LUNCH

Session II. Film Cooling Presentations

Chair: Steven Hippensteele

- 1:10 Film Cooling with Anisotropic Diffusion and Hole Entry Effects: Experiments and Computations
Terry Simon, University of Minnesota
- 1:30 Large Eddy Simulation and Improved Turbulence Modelling of Film Cooling
Sumanta Acharya, Louisiana State University
- 1:50 Heat Transfer in Film-Cooled Turbine Blades
Vijay Garg, AYT
- 2:10 Effect of Tabs on a Jet in Cross Flow
Khairul Zaman, NASA LeRC
- 2:30 Film Cooling Heat-Transfer Measurements Using Liquid Crystals
Steven Hippensteele, NASA LeRC
- 2:50 Turbine Vane Heat Transfer, Film Cooling, and Turbulence Experiments
Forrest Ames, Allison Engine Company
- 3:10 BREAK
- 3:30 Investigation of Rotor Wake Effects on Film Cooling
James Heidmann, NASA LeRC
- 3:50 Unsteady, Rotating Film-Cooling Experiments at Simulated Engine Conditions
Reza Abhari, Ohio State University
- 4:10 Unsteady High Tu Effects on Turbine Blade Film Cooling Heat Transfer Using Liquid Crystals
J. C. Han, Texas A&M
- 4:30 Group Discussion
- 5:30 Adjourn
- 5:30 Social
- 6:30 Dinner
Speaker: Michael Salkind, OAI President

December 13, 1996

Session III. Coolant Flow Integration & Optimization

Chair: *Joe Gladden*

- 8:30 Multi-Disciplinary Optimization of Cooled Turbine Design Using 3-D CFD Analysis
Aditi Chattopdhyay, Arizona State University
- 8:50 CFD/Exp. to Optimize Film-Cooling Hole Shape
Yong Kim, Allied Signal
- 9:10 Film Cooling Flow and Heat Transfer Design Tools
Robert Bergholz, General Electric Corporation
- 9:30 An Initial Multi-Domain Modeling of an Actively Cooled Structure
Erlendur Steinthorsson, ICOMP
- 9:50 Aero-Thermal-Structural Optimization
George Dulikravich, Penn State University
- 10:10 BREAK
- 10:30 Group Discussion
- 11:30 Closing Summary

REPORT DOCUMENTATION PAGE			Form Approved OMB No. 0704-0188	
Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.				
1. AGENCY USE ONLY (Leave blank)		2. REPORT DATE May 1997	3. REPORT TYPE AND DATES COVERED Technical Memorandum	
4. TITLE AND SUBTITLE Institute for Computational Mechanics in Propulsion (ICOMP) Eleventh Annual Report-1996			5. FUNDING NUMBERS WU-523-36-13-00 NCC3-370	
6. AUTHOR(S) Theo G. Keith, Jr., Karen Balog, and Louis A. Povinelli, editors				
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration Lewis Research Center Cleveland, Ohio 44135-3191			8. PERFORMING ORGANIZATION REPORT NUMBER E-10622	
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration Washington, D.C. 20546-0001			10. SPONSORING/MONITORING AGENCY REPORT NUMBER NASA TM-107476 ICOMP-97-01	
11. SUPPLEMENTARY NOTES ICOMP Program Director, Louis A. Povinelli, organization code 2600, (216) 433-5818.				
12a. DISTRIBUTION/AVAILABILITY STATEMENT Unclassified - Unlimited Subject Categories 34 and 64 This publication is available from the NASA Center for Aerospace Information, (301) 621-0390.			12b. DISTRIBUTION CODE	
13. ABSTRACT (Maximum 200 words) The Institute for Computational Mechanics in Propulsion (ICOMP) is operated by the Ohio Aerospace Institute (OAI) and funded under a cooperative agreement by the NASA Lewis Research Center in Cleveland, Ohio. The purpose of ICOMP is to develop techniques to improve problem-solving capabilities in all aspects of computational mechanics related to propulsion. This report describes the activities at ICOMP during 1996.				
14. SUBJECT TERMS Numerical analysis; Computer science; Mathematics; Fluid mechanics			15. NUMBER OF PAGES 51	
			16. PRICE CODE A04	
17. SECURITY CLASSIFICATION OF REPORT Unclassified	18. SECURITY CLASSIFICATION OF THIS PAGE Unclassified	19. SECURITY CLASSIFICATION OF ABSTRACT Unclassified	20. LIMITATION OF ABSTRACT	